

CFD SIMULATION OF TEMPERATURE DISTRIBUTION AND HEAT TRANSFER PATTERN INSIDE C492 COMBUSTION FURNACE

AMAR AL – FATHAH AHMAD

A thesis submitted in fulfilment for the award of the Degree of Bachelor in Chemical
Engineering (Gas Technology)

**Faculty of Chemical and Natural Resources Engineering
Universiti Malaysia Pahang**

APRIL 2009

I declare that this thesis entitled “CFD Simulation of Temperature Distribution and Heat Transfer Pattern inside C492 Combustion Furnace” is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature :

Name : Amar Al – Fathah Ahmad

Date : 21 APRIL 2009

*Dedicated, in thankful appreciation for support,
encouragement and understanding
to my beloved family and friends.*

ACNOWLEDGEMENT

In the name of Allah S.W.T. the most gracious and most merciful, Lord of the universe, with His permission Alhamdulillah the study has been completed. Praise to Prophet Muhammad S.A.W., His companions and to those on the path as what He preached upon, might Allah Almighty keep us His blessing and tenders.

First of all, I would like to express my gratitude to my research supervisor, Mr. Associate Professor Zulkafli Bin Hassan for his valuable guidance, advices, efforts, supervision and enthusiasm given throughout for the progress of this research. Although he always busy with his work, he can spend the time in completing this research.

I also wish to express my thankful to Mr. Mohd Masri A.Razak for conducting CFD class and gave so much guide and knowledge of the software. He has contributed towards my understanding and thoughts. Without the continued support and interest, this thesis would not have been the same as presented here.

I owe tremendous appreciation to my parents, Ahmad Md Isa and Siti Fatimah Ariffin for their support and motivation during the tough time throughout the duration of this study

Thank you so much for all. May Allah S.W.T. the Almighty be with us all the time.

ABSTRACT

Studies into the temperature distribution and heat transfer characteristics in a C492 combustion test furnace using commercial code FLUENT is presented in this paper. The mathematical model is based on an Eulerian description for the continuum phase and the model predicts gas flows, species concentrations and temperatures, particle trajectories and combustion and radiation heat fluxes. The gas phase conservation equations of momentum, enthalpy and mixture fraction are solved utilizing the k- ϵ turbulence model. Non premix combustion approach is used to predict the combustion process. The composition of the fuel component is setup using probability density function (PDF). A 3-D simplified model is created to determine the temperature and heat flux profiles and other thermal characteristics for a typical 150kW utility furnace firing liquid fuels or gaseous fuels. The temperature profiles of the furnace based on excess air ratio are predicted using the 3-D model. The main parameter is the excess air ratio consists of 1.248, 1.299, 1.362 and 1.417 which is used to study the temperature distribution. The model calculations showed a good agreement with the measured experimental data both in full and pilot scale of the test furnace as well as from the literature data. Using the experience gained from these CFD model studies can potentially improve the operation of a furnace, designing better combustion chamber or furnace with high performance and efficiency. Ultimately, these CFD model has the advantages of reduced cost, time and ability to optimize design significantly without much investment in the real experiment.

ABSTRAK

Kajian terhadap pengagihan suhu dan ciri – ciri pemindahan haba di dalam relau pembakaran C492 menggunakan kod FLUENT komersial dibentangkan di dalam kajian ini. Model matematik yang digunakan adalah berdasarkan Eulerian teori untuk fasa kontinuum dan digunakan untuk meramal pergerakan gas, kepekatan spesis dan suhu, trajektori partikel dan haba fluks pembakaran. Pemuliharaan persamaan momentum untuk fasa gas, entalpi, dan pecahan campuran diselesaikan menggunakan model pergolakan k- ϵ . “Non premix combustion” digunakan untuk meramal proses pembakaran di dalam simulasi ini. Komposisi bahan bakar ditetapkan menggunakan fungsi kebarangkalian ketumpatan (PDF). Model relau 3-D direka untuk menentukan suhu dan profil perbezaan haba serta lain – lain karakteristik thermal yang biasa untuk relau berkuasa 150kW yang menggunakan bahan api cecair atau gas. Profil - profil suhu bagi relau diramal berdasarkan kesan nisbah udara berlebihan ke atas model 3-D tersebut. Parameter utama digunakan adalah kesan nisbah udara berlebihan yang terdiri daripada 1.248, 1.299, 1.362 dan 1.417 dimana digunakan untuk mengkaji pengagihan suhu di dalam relau. Pengiraan model menunjukkan keputusan yang baik dimana menghampiri data – data dari eksperimen samada untuk relau di industri atau kajian kecil, malahan juga menghampiri data – data dari karya sastera. Pengalaman yang ditimba dari penggunaan kajian model CFD ini mampu meningkatkan operasi relau, mereka bentuk kebuk pembakaran atau relau yang lebih baik dan mempunyai kadar prestasi dan kecekapan yang tinggi. Akhirnya, model – model CFD ini mempunyai kelebihan dari segi pengurangan kos, masa dan kebolehan untuk memantapkan reka bentuk tanpa melibatkan eksperimen sebenar.

TABLE OF CONTENTS

CHAPTER	TITLE	PAGE
	ABSTRACT	v
	ABSTRAK	vi
	TABLE OF CONTENT	viii
	LIST OF TABLES	xi
	LIST OF FIGURES	xii
	LIST OF SYMBOLS	xiv
1	INTRODUCTION	
1.1	Research Background	1
1.1.1	Computational Fluid Dynamics	2
1.1.2	C492 Combustion Lab Unit	2
1.2	Problem Statement	3
1.3	Objectives	3
1.4	Scope of Study	4
1.5	Rationale and Significance	4

2**LITERATURE REVIEW**

2.1	Mathematical Modelling of NO _x Emission from High Temperature Air with Nitrous Oxide Mechanism	5
2.1.1	Effects of NO Model on the NO Formations	7
2.2	Summary of Previous Studies	9
2.3	Combustion Theory	9
2.3.1	Combustion Efficiency	11
2.4	Air Fuel Ratio	12
2.4.1	Effect of Excess Air Ratio	12
2.5	Liquefied Petroleum Gas (LPG)	13
2.6	Background of Study	14
2.6.1	Computational Fluid Dynamics (CFD)	14
2.6.1.1	Definition of CFD	14
2.6.1.2	History of CFD	15
2.6.1.3	Advantages of CFD	16
2.6.2	C492 Combustion Lab Unit	17
2.6.2.1	C492 Combustion Lab Unit Configuration	19

3**RESEARCH METHODOLOGY**

3.1	Overall Methodology	21
3.2	CFD Modelling and Simulation	22
3.3	Problem Solving Steps	23
3.3.1	Geometry Design/Meshing Geometry - GAMBIT	24

3.3.1.1	The Combustion Chamber Geometry and Materials	25
3.3.1.2	Combustion Chamber Meshing	25
3.3.2	FLUENT Simulation	27
3.3.2.1	CFD Computational Tools	27
3.3.2.2	CFD Governing Equations	28
3.3.2.2.1	Conservation Equations	28
3.3.2.3	Turbulence Modelling	29
3.3.2.4	FLUENT Discretisation	29
3.4	Experimental Methodology	31
3.4.1	Experimental Methodology Outline	32
3.4.2	Materials and Equipments	33
3.4.3	Operating Procedure	33
3.4.4	Experimental Procedure	34
4	RESULTS AND DISCUSSIONS	
4.1	Introduction	35
4.2	Experimental Results	36
4.3	Simulation Results	37
4.3.1	Temperature Distribution at Side View of Geometry	37
4.4	Comparison Between Simulation and Experimental Results	39
5	CONCLUSION AND RECCOMENDATIONS	
5.1	Conclusion	42
5.2	Recommendations	43

REFERENCES	44
APPENDIX A: CFD COMPUTATIONAL TOOLS	47
APPENDIX B: FUEL PROPERTIES	64
APPENDIX C: FLUENT SIMULATION RESULTS	69
APPENDIX D: EXPERIMENTAL DATA	81

LIST OF TABLES

TABLE NO.	TITLE	PAGE
2.1	Summary of Literature Review	9
3.1	Boundary Conditions	30
4.1	Experimental Results	36
4.2	Comparison of Flue Temperature	40
4.3	Comparison of Flame Temperature	40

LIST OF FIGURES

FIGURE NO.	TITLE	PAGE
2.1	Effects of excess air ratio on NO emission	6
2.2	Effect of excess air ratio on maximum temperature	7
2.3	Temperature distribution at the cross section through fuel and one air nozzle in the test furnace at different excess air ratio (λ)	8
2.4	C492 Combustion Lab Unit	18
2.5	Control Panel of C492 Combustion Lab Unit	18
2.6	Schematic Diagram of Gas Burner fitted on C493 Combustion Lab Unit	19
3.1	Flow charts of research methodology	22
3.2	Program Structure	23
3.3	Steps of CFD analysis	24
3.4	Overall combustion chamber dimension	25
3.5	Meshed combustion chamber geometry	26

3.6	Iteration process	31
3.7	Flow chart of experimental methodology	33
4.1	Temperature distribution at cross section through fuel and air nozzle in furnace at different excess air ratio (λ) at level 20 distribution	38
4.2	Flame cross section through fuel and air nozzle in furnace at different excess air ratio (λ) at level 3 distribution	38
4.3	Velocity vector based on total temperature of the furnace.	39
4.4	Effect of excess air ratio on flue temperature and flame temperature	41

LIST OF SYMBOLS

LPG	-	Liquefied Petroleum Gas
CFD	-	Computational Fluid Dynamics
CAD	-	Computer aided design
CAE	-	Computer aided engineering
RANS	-	Reynolds averaged Navier Stokes
OD	-	Outside diameter
ID	-	Inside diameter
CO ₂	-	Carbon dioxide
NO _x	-	Oxides of Nitrogen
AFR	-	Air fuel ratio
O ₂	-	Oxygen
kW	-	Kilowatts
K	-	Kelvin
CH ₄	-	Methane
C ₂ H ₆	-	Ethane
C ₃ H ₈	-	Propane
C ₄ H ₁₀	-	Butane
F	-	Force
m	-	mass
a		acceleration

PIC	Particle in cell
MAC	Marker and Cell
ALE	Arbitrary Lagrangian Eulerian
PDF	Probability density function
mm	millimetre
kg	kilogram
V	volt
n_c	Combustion efficiency
λ	Excess air ratio

CHAPTER 1

INTRODUCTION

1.1 Research Background

Space heating is very important in industrial sector. It emphasis on effective heat transfer, and heat is supplied through furnace, burner or combustion chamber. Therefore it is important to determine what going on inside the furnace especially the heat transfer pattern. Heat transfer inside the combustion chamber is governed by the temperature distribution from the flame and the product concentration [1]. For heat to be transferred effectively, the temperature must be distributed uniform throughout the chamber, as the flame give out the heat. However, inside the chamber, there are certain areas that have temperature lower than the distributed temperature. This is due to the presence of NO_x inside the chamber which converts the useful heat into waste heat [2]. Furthermore, the NO_x is a pollutant and can cause corrosion with presence of water vapour. When overall heat transfer is lowered, the combustion efficiency of the furnace or combustion chamber is decreased [3, 4]. However, combustion efficiency is related to the air to fuel ratio, when insufficient air supplied will cause higher amount of uncombusted fuel goes into hydrocarbon emission which lowers efficiency [3, 4]. In other hand, too much air will cool the combustion chamber and carry a larger percentage of the heat out of the flue, reducing combustion efficiency [3, 4]. Therefore, it is important for to know how the temperature are distributed in the event of such conditions which later can be studied in order to achieve better design for furnace and combustion chamber.

1.1.1 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is a part of fluid mechanics that applies numerical methods and algorithms in order to solve and analyse problems that relates to flows of fluid. In order to done the calculations, computers are used to compute such task by using specific software that allows complex calculation of a simulation of intended flow process [5]. Using CFD, we can build a computational model that represents a system or device that we want to study. In our case, CFD – FLUENT is used which is a software that uses the science of predicting fluid flow, heat and mass transfer, chemical reactions and related phenomena by solving numerically the sets of governing mathematical equations. The results of CFD analysis are relevant in conceptual studies of new design, detailed product development, troubleshooting and redesigning. Therefore, by using CFD FLUENT, we can build a 'virtual prototype' of the system or device that we wish to analyse and then apply real-world physics and chemistry to the model, and the software will provide with images and data, which predict the performance of that design.

1.1.2 C492 Combustion Lab Unit

The combustion chamber that are used to gather the experimental data for the simulation is C492 Combustion Lab Unit which enables studies into many aspects of combustion and burner operation using the optional burners or any suitable commercially available oil or gas burner up to 150 kW. The four large observation windows fitted in the frame mounted, water cooled, combustion chamber provide an excellent flame demonstration facility. The full instrumentation and safety features allow supervised student operation over a wide range of air/fuel ratios and different fuels [6].

1.2 Problem Statement

Inside today's industrial furnace and combustion chamber, there a lot of uncertainties inside the chamber, especially on flame and temperature distribution. As we all know, air to fuel ratio plays a very important role in determining the combustion efficiency. Too little or too much air will lower the overall combustion efficiency. However, the ones that we do not fully understand are how the temperature is distributed inside the furnace. Therefore, it is important to know how the temperature is distributed inside the chamber and learns how the air to fuel ratio affect the temperature distribution. Hence, we are required to validate the experimental data and compared to the result of simulation through CFD. By validating data from both sets, we can determine the correct methods and models that allows us to implement new design without spent much cost by using CFD simulation software which can reduces the time and operation cost compared to experiments setup and prototyping.

1.3 Objectives

The objectives of this study are:

1. To develop a combustion chamber geometry model using CFD FLUENT in order to simulate the flame and temperature profile inside combustion lab unit C492
2. To validate the simulation results with the experimental results.

1.4 Scope of Study

The scopes of this study are:

1. The scope of this project is to run simulations on temperature distribution and heat transfer pattern inside C492 Combustion Chamber. The limitation of our project is fuel type which is liquid and gaseous fuel. LPG is chosen as only fuel used in the experiment because of majority of today furnaces and combustion chambers use LPG as fuel source. Moreover, the type of LPG used mainly was based on United States' HD – 5 standards. The main compositions of (mass %) were $\text{CH}_4=0.02\%$, $\text{C}_2\text{H}_6=0.95\%$, $\text{C}_3\text{H}_8=98.35\%$, $\text{C}_4\text{H}_{10}=0.67\%$. For the case study, the fuel inlet temperature and air inlet temperature is at ambient temperature of 296 K [7]
2. The simulation model geometry is designed in simplified geometry which is enough to cover the intended objectives. As for the flow and mass transfer, the balance equations used are standards in the CFD – FLUENT 6 and used the Reynolds Average Navier – Stokes (RANS) formulation [5, 8]. Furthermore, the fluid or the fuels are assumed to be Newtonian, and to follow the perfect gas law. The $k - \epsilon$ are used as closure equations and standards value for constants are adopted. [5, 8]. The state of the overall process is considered to be steady state process.
3. To validate the simulation result, we compared the composition of flue gas from the experiment with the simulation data. The simulation result is expected to converge within acceptable error of the actual result.

1.5 Rationale and Significance

The importance of this study is to validate the simulation results with the actual experimental results. If the simulation results fit the actual experimental results, it means that we can test our objectives through simulation instead of doing the expensive actual experiment. This enables us to forecast and analyse data before put to the actual experiment. Therefore, we can significantly improve the design and reduce cost for fabrication of prototype.

CHAPTER 2

LITERATURE REVIEW

This chapter provide a literature reviews with relations to the past research effort such as journals or articles related to combustion process, effect of air/fuel ratio on combustion efficiency of LPG fuel and computational fluid dynamics (CFD) analysis whether on two dimension and three dimension modelling. Moreover, reviews on other relevant studies are made in order to relate to our project.

2.1 Mathematical Modelling of NO_x Emission from High Temperature Air with Nitrous Oxide Mechanism

Previous studies show that the modelling of NO_x emission in term of excess air ratio and temperature distribution. Figure 2.1 depicts the results from experimental measurement and simulation prediction. The experimental results were taken at the burner outlet and the chimney of the furnace. The graph is based on effect of excess air ratio which consists of actual measured results, predicted simulation result with N₂O route and predicted simulation result without N₂O route. The simulation result without the N₂O route is lower than both of measured and prediction with N₂O route. The reason is that when the N₂O route is not present, the simulation assumed that mixing rate was considered complete at excess air ratio equal to 1.04 which allows no formation of NO. As the excess air ratio increases, the temperature increases to which point it allow the simulation to predict the value of NO emission. The simulation with N₂O route shows that NO was formed at lower excess air which corresponds to the measured experimental result which shows NO

formation at every excess air ratio. When comparing with experimental data, the model with N_2O route shows a very good agreement when excess air ratio is equal or less than 1.15. On the other hand, when excess air ratio is equal to 1.25, NO emission is almost two times higher than experimental data. [9]

The result is related to Figure 2.2 which shows the experimental data on the effect of excess air ratio on the maximum temperature, the graph shows that with increasing amount of excess air ratio will increase the maximum temperature of the furnace. It is in fact that oxygen concentration in reaction zone increases as excess air ratio increase which results in a higher flame temperature [9]. It is worth to note that the furnace model used was semi industrial furnace with HiTAC system compared to the one used in this study. Nonetheless, the parameter applies to furnace and combustion chamber in general. Therefore, the results from the literature can be used as base for this project.

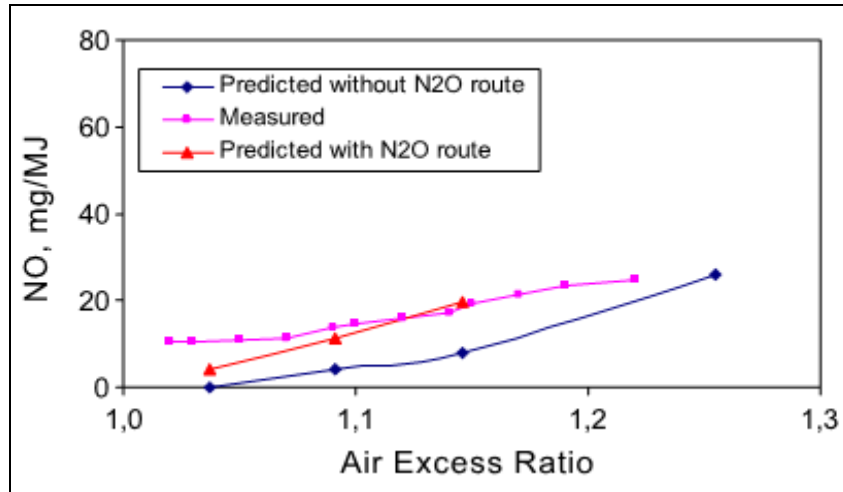


Figure 2.1: Effects of excess air ratio on NO emission.

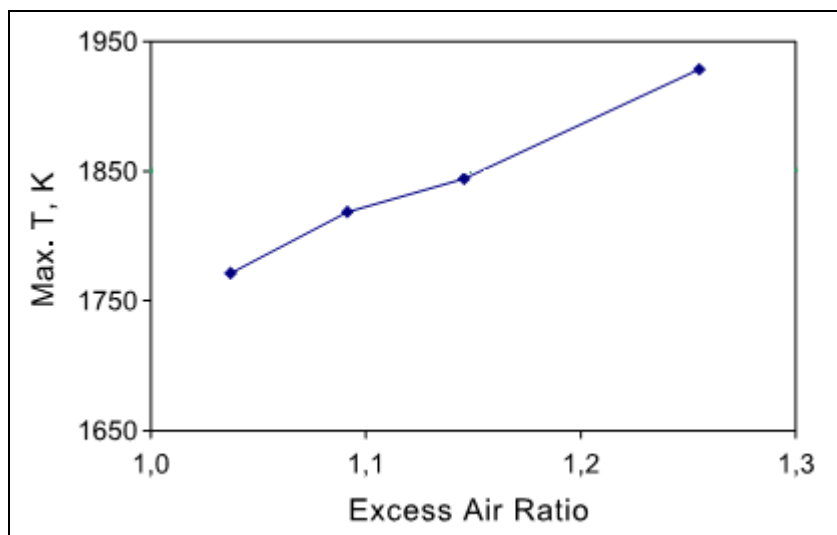


Figure 2.2: Effect of excess air ratio on maximum temperature.

2.1.1 Effects of NO Model on the NO Formations

In order to further understand the effects of the NO model on NO formation and destruction inside furnace or combustion chamber is by investigating temperature profiles as shown in Figure 2.3 which the cross section of the furnace at different excess air ratio. There is usually large chemical reaction zone in all case which can be verified by experimental studies. The highest temperature usually occurs at the middle of the furnace which is where the flame situated. This literature uses HiTAC burner which mix the fuel and combustion air at initial stage of the combustion which also the same operation for the C492 gas burner that is used in this study. The combustion product is entrained into the root of flame because of high injection momentum which reduces the oxygen availability in the primary combustion zone. The unburnt fuel that escaped the primary combustion zone gradually mixed with air to complete the combustion, resulting in more uniform temperature profile. So that, the production rate of NO is lowered. [9]

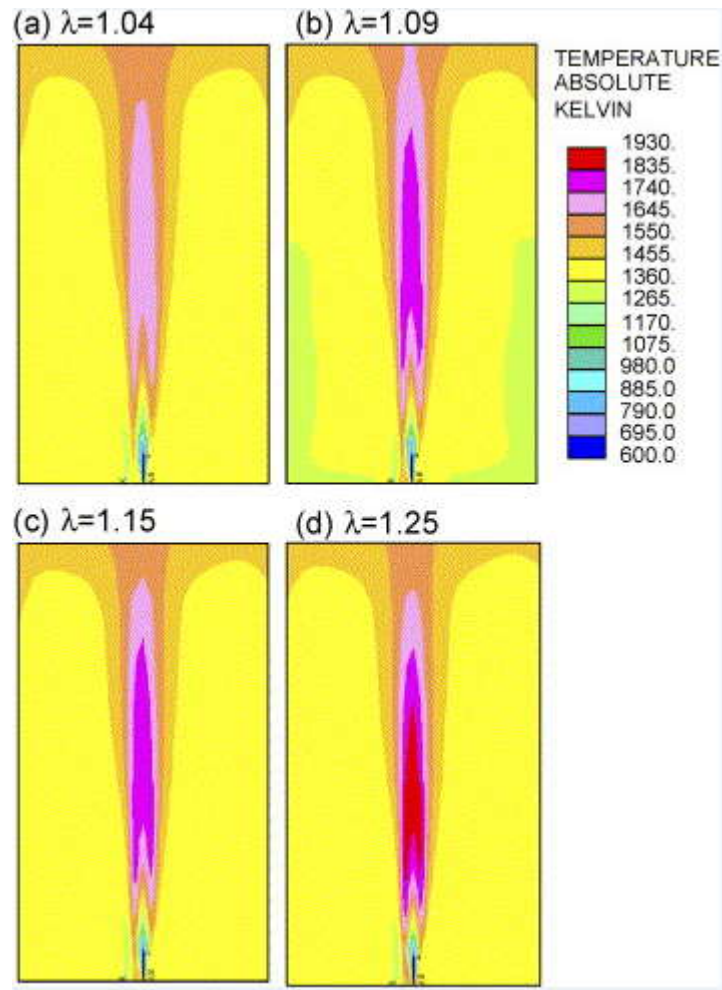


Figure 2.3: Temperature distribution at the cross section through fuel and one air nozzle in the test furnace at different excess air ratio (λ).

The literature concludes that they developed the N_2O route in order to predict the NO formation and emission inside combustion chamber. The results show that the NO emission formed by N_2O intermediate mechanism is important in furnaces that have low peak temperature. Moreover, the increasing excess air ratio leads to increasing NO emission under high temperature air combustion condition.

2.2 Summary of Previous Studies

Table 2.1 below summarized the findings and review others previous work which related to this research.

Table 2.1: Summary of Literature Review

Authors and Year	Study Parameter	Results and Findings
Weihong Yang, and Wlodzimierz Blasiak., 2005[10]	1. Properties of LPG flame 2. Effect of air/fuel ratio on test furnace	1. Size and shape matches the observed flame in test furnace 2. Increasing excess air ratio leads to increasing NO emission
Weihong Yang, and Wlodzimierz Blasiak., 2005[9]	1. NOx emission profile 2. Effect of excess air ratio on NOx emissions	1. Lower excess air ratio leads to larger flame volume and larger entrainment ratio, thus reducing NO emissions
Nabil Rafidi, and Wlodzimierz Blasiak., 2006[11]	1. Heat transfer characteristic inside test furnace 2. Temperature distribution profile	1. Enhancement of temperature and heat flux uniformity. 2. Uniform temperature distribution reduce NOx level inside test furnace

2.3 Combustion Theory

Combustion is defined as burning or in chemical terms as rapid mixture of flammable substance with oxygen which accompanied by the evolution heat of in form of light radiation. Combustion process is an exothermic reaction which is the heat energy from the reaction is given out. As clear example of combustion is a slow burning of candle flame and the explosion of petrol vapour in air [12].

There are six types of combustion process, which are rapid, slow, complete, turbulent, microgravity and incomplete combustion. In rapid combustion, the process releases high amount of heat and energy. In addition, some large amount gaseous will also be produced from the combustion resulting in excessive pressure build-up which

eventually create high noise event called explosion [12]. On contrary, slow combustion occurs at very low temperature which often correlates to the cellular respiration as example. Turbulent combustion is a process where the combustion process is characterized by turbulent flow which helps the mixing between the oxidizer and the fuel producing complete combustion. On other part, microgravity combustion is a combustion process that occurs at low gravity or no gravity condition and often produces spherical flame which is important in understanding combustion physics in space [12].

Complete combustion is a process where the reactant will burn completely in air or oxygen producing limited number of products. In common sense, hydrocarbons which consists of hydrogen and carbon combust with oxygen producing carbon dioxide and water. It's also applies to elements such as nitrogen, sulphur and iron which when are burned will yield common oxides [12]. Complete combustion is also referred as stoichiometric combustion which the theoretical air supplied at exact amount.

Incomplete combustion occurs when air or oxygen supplied is not enough for reactant to completely consume in the reaction or when heat sinks quenched the combustion process. When incomplete combustion happens, the part of the reactant remained unoxidized which will produce additional or unwanted combustion products which mostly are pollutants. The pollutants that may arise from such conditions are carbon monoxides, nitrogen oxides or even sulphur oxides. The increased level of the pollutants means that the flue gas from the combustion can be dangerous and toxic to living things. Moreover, incomplete combustion greatly reduces the heat transfer process and reduces overall combustion efficiency [12].

In actual practice, air is supplied in excess of that of theoretical requirement in order to ensure complete combustion. Excess air can be expressed as percentage of theoretical air needed i.e. 10% excess air is equivalent to 1.1 times of theoretical air quantity. By supplying excess air, the quantity of fuel being wasted is reduced and the variations in fuel quality or air can still be tolerated and guaranteed a complete combustion [4].

2.3.1 Combustion Efficiency

The key to combustion efficiency is the control of excess air ratio. In which when too little will cause incomplete combustion with the pollutants emission, whilst too much will cool the combustion chamber and carry a larger percentage of the heat out of the flue, reducing combustion efficiency. Therefore, combustion efficiency is a calculation of how well your equipment is burning a specific fuel, shown in percent. Complete combustion efficiency would extract all the energy available in the fuel. However 100% combustion efficiency is not realistically achievable. Common combustion processes produce efficiencies from 10% to 95%. Combustion efficiency calculations assume complete fuel combustion and are based on three factors: [3]

1. The chemistry of the fuel.
2. The net temperature of the stack gases.
3. The percentage of oxygen or CO₂ by volume after combustion.

$$n_c = f(\text{airflow rate})^{-1} (1/\text{evaporation rate} + 1/\text{mixing rate} + 1/\text{reaction rate})^{-1}$$

Combustion efficiency also can be expressed as above as the total time required to burn a liquid fuel is the sum of the times required for fuel evaporation, mixing of fuel vapour with air and combustion products, and chemical reaction which relates that time available for combustion is inversely proportional to the airflow rate [3].

2.4 Air Fuel Ratio

The surrounding air consists of 21% oxygen that play major role in combustion and 79% nitrogen which inert that takes no part in combustion process at low temperature. Air is the primary sources of oxygen which is plenty but when it enters combustion process, due to presence of nitrogen, much of the heat of the process is wasted. This is a basic inefficiency which has to be accepted as there is no

economical way of separating the oxygen and nitrogen. Therefore, the ratio of which air and fuel is to be taken into combustion process very important in which it determines the overall efficiency. Air-fuel ratio (AFR) can be defined as the mass ratio of air to fuel present during combustion. When all the fuel is combined with all the free oxygen, typically within a vehicle's combustion chamber, the mixture is chemically balanced and this AFR is called the stoichiometric mixture. AFR is an important measure for anti-pollution and performance tuning reasons. Lambda (λ) is an alternative way to represent AFR. [4]

2.4.1 Effect of Excess Air Ratio

All hydrocarbon fuel has a condition of perfect combustion, in which all the oxygen in the air is consumed, but without excess air. In practice, to ensure complete combustion, excess air is supplied beyond that theoretically required for full oxidization of the fuel. This express as a percentage of the theoretical air needed i.e. 10% excess air is 1.1 times the theoretical air quantity. Having excess air ensures no fuel is wasted, and variations in fuel quality or air and fuel rates can be tolerated and still guarantee complete combustion. The stoichiometric air/fuel ration is different for each fuel because it depends on its chemical composition. In Appendix B, the table of stoichiometric condition of typical fuels is described. Therefore, in the combustion chamber, the effect of excess air ratio plays very important role in combustion efficiency.

2.5 Liquefied Petroleum Gas (LPG)

Liquefied Petroleum Gas (LPG) is a mixture of hydrocarbon gases which primarily consists of butane, propane or methane in certain ratio. LPG is used as fuel sources in heating appliances and vehicles. Common LPG ratios are 60% propane and 40% butane which often depend on season change. Other components in LPG are odorant such as ethanethiol and small concentration of propylene or butylenes.

LPG is synthesized by refining petroleum or extracted from wet natural gas. Wet natural gas meant that the natural gas still contain high amount of heavier hydrocarbons than methane. Although LPG is not renewable fuel, however its advantages weighed more and becoming more popular choice presently. The advantages of LPG are its burns cleanly with no soot or sulphur emission, posing no ground or water pollution hazard and it contains much higher heating value than diesel or gasoline. LPG has a typical specific caloric value of 46.1 MJ/kg compared to 42.5 MJ/kg for diesel and 43.5 MJ/kg for gasoline. However, LPG's energy density per unit volume is much lower than diesel and gasoline which correlates to the volume that LPG can be stored in same container in comparison of diesel and gasoline [7].

The commercialized LPG is in liquid state inside pressurized steel containers or cylinders because at normal temperature and pressures, the LPG will evaporate. However, the LPG is not filled completely in its container to allow thermal expansion of the gases in between 80% to 85% of their filled capacity. The common volume ratio between vaporised gas and the liquefied gas is about 250:1 but it also varies depend on composition, pressure, and temperature. The pressure inside the container is called vapour pressure which depends on LPG composition and pressure i.e. approximately 2.2 bar for pure butane at 68°F and 22 bar for pure propane at 131°F. The density of LPG is much higher than air which means it will settle down on the ground which causes dangerous hazards such as suffocation or explosion if there is source of ignition. [7]

2.6 Background of Study

This section describes the background of study which consists of the explanation on software and equipment used to complete this project. The software principle is presented along with information regarding to the software. The equipment used to done the experiment also is presented.

2.6.1 Computational Fluid Dynamics (CFD)

Computational fluid dynamics is part of fluid mechanics that uses numerical method and algorithm to solve and analyse problem that related to fluid flow. This section describes the definition, history and the advantage of using computational fluid dynamics.

2.6.1.1 Definition of CFD

CFD is a computer-based mathematical modelling tool that incorporates the solution of the fundamental equations of fluid flow, the Navier-Stokes equations, and other allied equations. CFD incorporates empirical models for modelling turbulence based on experimentation, as well as the solution of heat, mass and other transport and field equations. In order to done the calculations, computers are used to compute such task by using specific software that allows complex calculation for simulation of intended flow process. There are three phases to CFD: pre-processing, or creation of a geometry usually done in a CAD tool; mesh generation of a suitable computational domain to solve the flow equations on; and solving with post processing, or visualization of a CFD code's predictions. CFD is now a widely accepted and validated engineering tool for industrial applications. In our case, CFD – FLUENT is used which is a software that uses the science of predicting fluid flow, heat and mass transfer, chemical reactions and related phenomena by solving numerically the sets of governing mathematical equations. The results of CFD analysis are relevant in conceptual studies of new design, detailed product development, troubleshooting and redesigning [5, 8].

2.6.1.2 History of CFD

It is debatable as to who did the earliest CFD calculations (in a modern sense) although Lewis Fry Richardson in England (1881-1953) developed the first numerical weather prediction system when he divided physical space into grid cells and used the finite difference approximations of Bjerknes's "primitive differential equations". His own attempt to calculate weather for a single eight-hour period took six weeks of real time and ended in failure! His model's enormous calculation requirements led Richardson to propose a solution he called the "forecast-factory". The "factory" would have involved filling a vast stadium with 64,000 people. Each one, armed with a mechanical calculator, would perform part of the flow calculation. A leader in the centre, using coloured signal lights and telegraph communication, would coordinate the forecast. What he was proposing would have been a very rudimentary CFD calculation.

During the 1960s, the theoretical division of NASA at Los Alamos in the U.S. contributed many numerical methods that are still in use in CFD today, such as the following methods: Particle-In-Cell (PIC), Marker-and-Cell (MAC), Vorticity-Stream function methods, Arbitrary Lagrangian-Eulerian (ALE) methods, and the ubiquitous $k - \epsilon$ turbulence model. In the 1970s, a group working under D. Brian Spalding, at Imperial College, London, developed Parabolic flow codes (GENMIX), Vorticity-Stream function based codes, the SIMPLE algorithm and the TEACH code, as well as the form of the $k - \epsilon$ equations that are used today (Spalding & Launder, 1972). They went on to develop Upwind differencing, 'Eddy break-up' and 'presumed PDF' combustion models. Another key event in CFD industry was in 1980 when Suhas V. Patankar published "Numerical Heat Transfer and Fluid Flow", probably the most influential book on CFD to date, and the one that spawned a thousand CFD codes.

It was in the early 1980s that commercial CFD codes came into the open market place in a big way. The use of commercial CFD software started to become accepted by major companies around the world rather than their continuing to develop in-house CFD codes. Commercial CFD software is therefore based on sets of very complex non-linear mathematical expressions that define the fundamental

equations of fluid flow, heat and materials transport. These equations are solved iteratively using complex computer algorithms embedded within CFD software. The net effect of such software is to allow the user to computationally model any flow field provided the geometry of the object being modelled is known, the physics and chemistry are identified, and some initial flow conditions are prescribed. Outputs from CFD software can be viewed graphically in colour plots of velocity vectors, contours of pressure, lines of constant flow field properties, or as "hard" numerical data and X-Y plots.

CFD is now recognized to be a part of the computer-aided engineering (CAE) spectrum of tools used extensively today in all industries, and its approach to modelling fluid flow phenomena allows equipment designers and technical analysts to have the power of a virtual wind tunnel on their desktop computer. CFD software has evolved far beyond what Navier, Stokes or Da Vinci could ever have imagined. CFD has become an indispensable part of the aerodynamic and hydrodynamic design process for planes, trains, automobiles, rockets, ships, submarines; and indeed any moving craft or manufacturing process that mankind has devised. [13]

2.6.1.3 Advantages of CFD

Nowadays, CFD simulation software is widely used across the world in many sectors. Wide range of application is used in CFD to simulate in the lot of the aerospace, nuclear and automotive sectors through manufacturing, chemical and process sectors to the pharmaceuticals, biomedical and electronics industries.

The advantages using this software:

1. This software allows us to make changes to the analysis at any time during the setup, solution, or post processing phase. This saves time and enables us to refine our designs efficiently. The intuitive interface makes learning easy. Smart panels show only the modelling options that are appropriate for the problem setup at hand. Computer aided design (CAD) geometries are easily

imported and adapted for CFD solutions.

2. Solver enhancements and numerical algorithms that decrease the time to solution are included in every new release of our software. Our mature, robust, and flexible parallel processing capability enables us to solve bigger problems faster, and has been proven on the widest possible variety of platforms in the industry.
3. FLUENT's post processing provides several levels of reporting, so we can satisfy the needs and interests of all audiences. Quantitative data analysis can be as rigorous as we require. High resolution images and animations allow us to communicate our results with impact. Numerous data export options are available for integration with structural analysis and other computer aided engineering (CAE) software programs.

2.6.2 C492 Combustion Lab Unit

This unit enables studies into many aspects of combustion and burner operation using the optional burners or any suitable commercially available oil or gas burner up to 150 kW. The four large observation windows fitted in the frame mounted, water cooled, combustion chamber provide an excellent flame demonstration facility. The full instrumentation and safety features allow supervised student operation over a wide range of air/fuel ratios and different fuels [6]. Figure 2.4 depicts the picture of the combustion chamber.



Figure 2.4: C492 Combustion Lab Unit [6]



Figure 2.5: Control Panel of C492 Combustion Lab Unit [6]

2.6.2.1 C492 Combustion Lab Unit Configuration

The C492 Combustion Lab Unit used in this study is equipped with two interchangeable one-flame burner systems which is one is for liquid fuel and the another is for gaseous fuel. The overall dimension of the Lab Unit are 1560mm(height) x 800mm(width) x 1800mm(depth) and the weight are 250kg for 220V unit(260kg for 110V unit)[6]. The frame is powder coated 25mm mild steel angle frame with 16 swg shelves on which the combustion chamber is mounted. The stainless steel chamber is mounted horizontally on the frame and has a water jacket around the outside and at the rear end through which mains water passes to remove heat produced from the burner. At the front end is the burner mounting plate which has studs onto which either gas or oil burners can be mounted. The inside of the plate has insulation to prevent excessive heat exposed to the operator's position. There are 2 circular high temperature glass windows along side the chamber for which the flame can be observed.

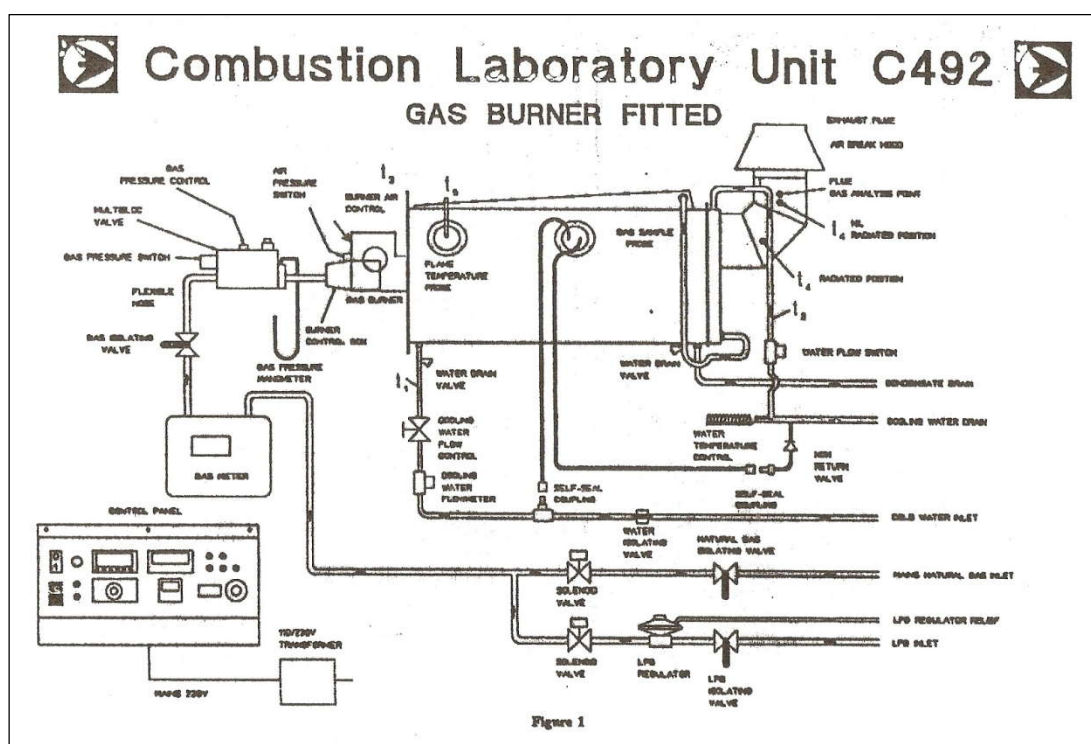


Figure 2.6: Schematic Diagram of Gas Burner fitted on C493 Combustion Lab Unit.

The dimensions of the stainless steel combustion chamber are 450mm ID @ 500mm OD x 1000mm long with four 100mm diameter quartz glass window. The front plate has 80mm thick insulation and burner mounting stud. At the rear end, there is one flue gas duct of 150mm OD for removing the hot flue gas from the combustion chamber which goes through an insulated 90° bend section. At this section, there are tapping points which exhaust thermocouple, air and gas analyser sensor probe can be inserted. An air break hood is installed to the flue duct to entrain ambient air and to cool the flue gases. Condensate that may collect within the chamber is drained away from a point towards the rear of the chamber [6].

A 150kW one-flame C492A Gas Burner is used in this study. The one flame gas burner features one fuel nozzle surrounded by air inlets and fuel gas outlet. The fuel and the air is injected directly into the combustion chamber at high velocities which the strong entrainment led by these injection moment will cause the combustion air to be well mixed with the fuel.

CHAPTER 3

METHODOLOGY

3.1 Overall Research Methodology

The flow charts in Figure 3.1 shows the overall research methodology of this project. Numerical approach will be used to solve the three dimensional analysis of C492 Combustion Lab Unit. The boundary condition is taken from the experimental data. Excess air ratio is the main parameter data will be validated at the early stages and the effect of excess air ratio on combustion efficiency will be directly compared on later stages.

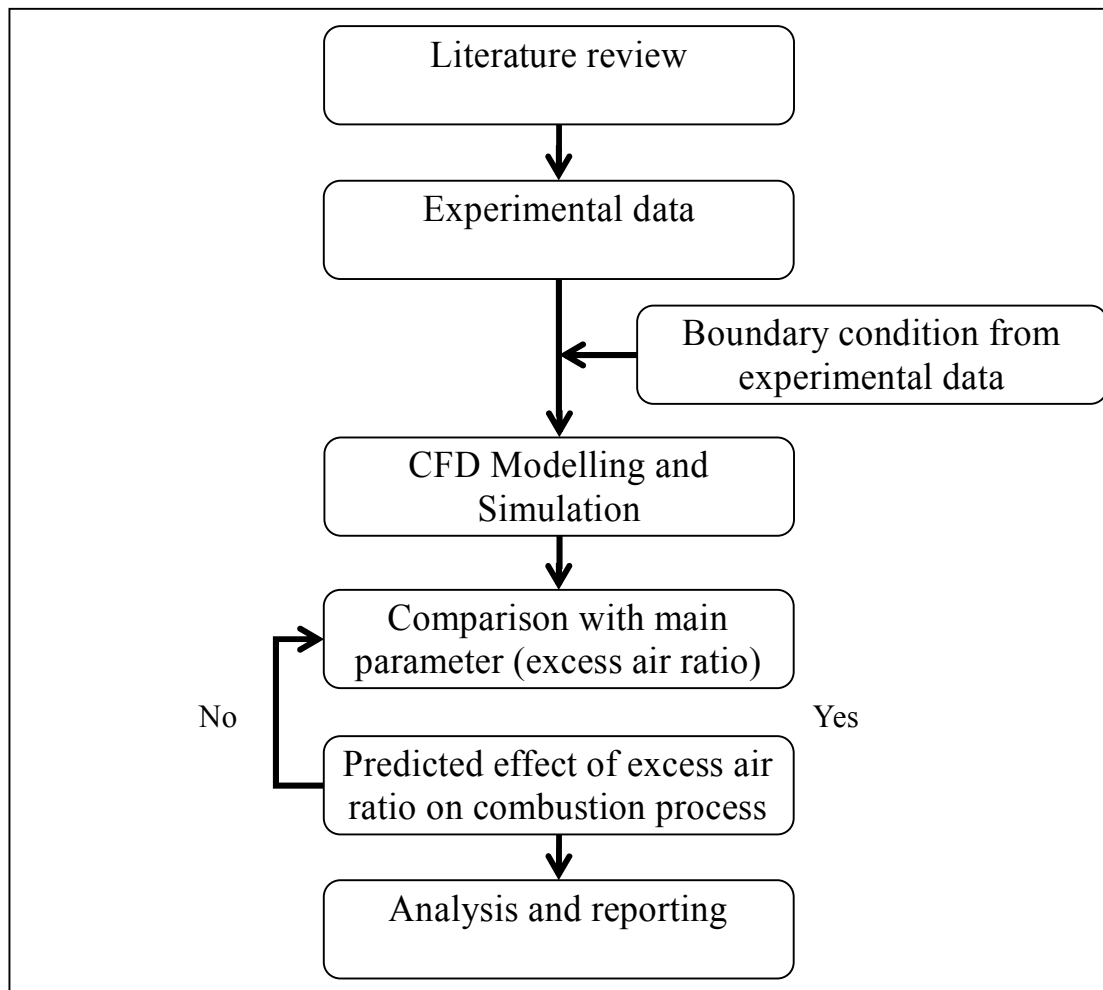


Figure 3.1: Flow charts of research methodology.

3.2 CFD Modelling and Simulation

CFD – FLUENT 6 is used for the modelling and simulation in this project. CFD – FLUENT 6 is computer software that allows modelling and simulation of flow of fluid and heat and mass transfer in complex geometries. It is capable to complete meshing flexibility, solving flow problems with unstructured meshes that can be generated through the complex geometries. The program structure is shown in Figure 3.2.

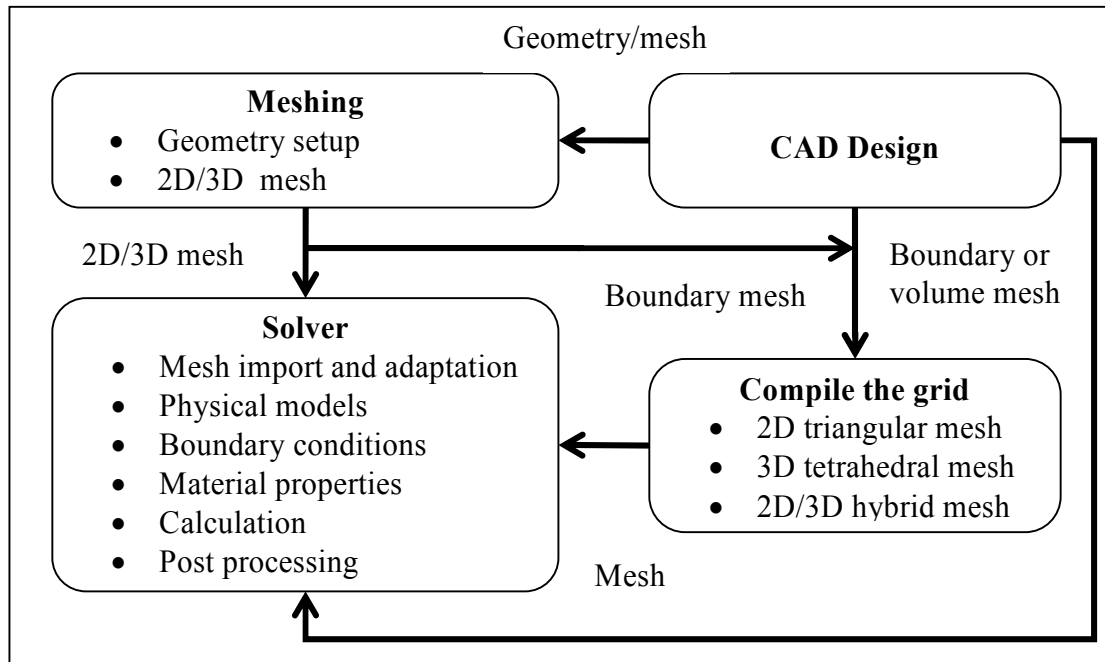


Figure 3.2: Program Structure.

3.3 Problem Solving Steps

By using the CFD – FLUENT 6, the problem solving steps involves the problem identification, grid creation, solver execution and analysis of the result. The advantages is given by its build in post processing that enables automatic save on setting file data written in *.dat or *.cas file. The detailed steps are shown in Figure 3.3.

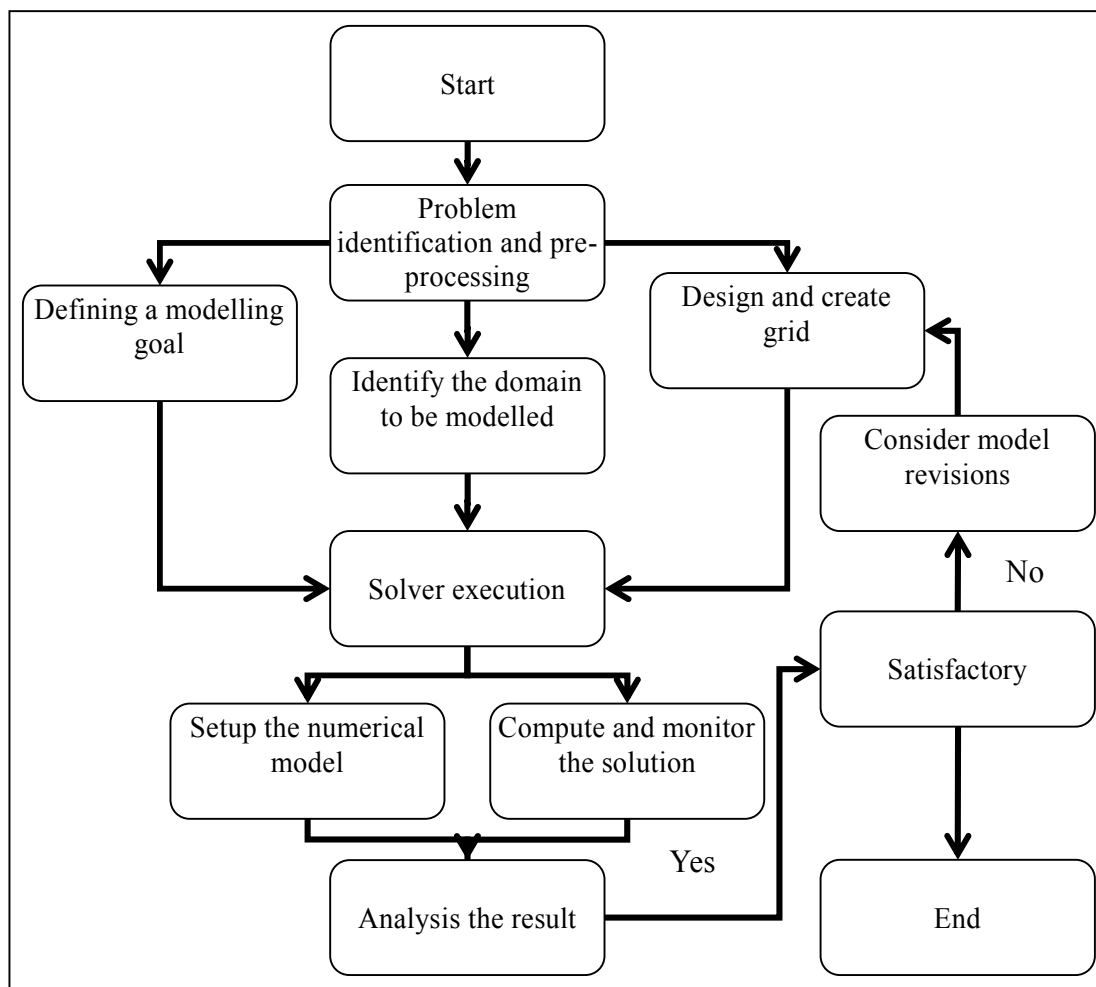


Figure 3.3: Steps of CFD analysis.

3.3.1 Geometry Design/Meshing Geometry – GAMBIT

In order to create a suitable model to run in FLUENT software, pre-design geometry must be created to match with the needs of FLUENT. Therefore, GAMBIT which is a geometric modelling and grid generation tool is provided along with the FLUENT technology. GAMBIT allows the user to import geometry from other designing software or computer-aided design (CAD) software or create own geometry entirely based on GAMBIT itself. In addition, GAMBIT can automatically mesh surfaces and volume while allowing the user to manipulate the mesh through size functions and boundary layer meshing.